Numerical Investigation of Fluid Flow Through A 2D Backward Facing Step Channel

Mohammad A. Hossain, Md. Taibur Rahman, Shakerur Ridwan
Department of Mechanical Engineering
The University of Texas at El paso
El paso, TX, USA

ABSTRACT

The work is focused on numeric investigation of fluid flow through a 2D backward facing step channel with an expansion ratio of 1:2. Water is used as a working fluid. Commercial software ANSYS Fluent has been used to resolve the flow field especially the separation length. Measurement has been taken to find the reattachment length at different Reynolds number ranging from 100 to 4000. The simulated data have been compared with available experimental data. It has been observed that the simulated data follows the same pattern as the experiment. Horizontal and vertical velocity component have been calculated at different Reynolds number and presented. Streamline shows the flow separation at different location of the channel. It has been observed that as the inlet velocity increases, reattachment length also increases at a certain range than it starts to fall as velocity increases.

Key word: backward step, flow separation, reattachment

INTRODUCTION

Flow through a backward facing step is one of the classical internal flow problem in fluid mechanics. Extensive numerical and experimental studies have been done on this issue. Due to the geometric simplicity and flow diagnostic technique, this phenomena has become a practice for numeric validation for computational fluid dynamics (CFD). In this paper fluid flow through a backward facing step has been studied numerically for a 2D channel having an expansion ration (H/h) = 2. Commercial software package ANSYS Fluent has been used for the numeric simulation. Different flow characteristics has been calculated at different Reynolds number (Re = 100 to 4000) in order to get the actual flow behavior. Calculations are done for different flow regime such as laminar, transient and turbulent. Finally the observed data such as horizontal velocity, separation length, etc have been compared with available experimental data.

A significant amount of work on flow through backward facing step has been done by Armaly et al [1]. They did a detailed analysis of flow behavior inside a backward facing step channel experimentally and numerically. Ample amount of work have been done by numerous people. Lee and smith [2] used the potential flow theory to address the problem. The potential flow theory did not able to predict the flow separation or reattachment region behind the step. Early numeric predictions of backward facing step flows were done by Roache [3], Taylor and Ndefo [4] and Durst and Pereira [5]. A separation region at the lower corner of the step was predicted by Alleborn et al [6] after a careful study of a sudden expansion of such channel. Brandt et al [7] and Hack bush [8] used multigrid method and Lange et al [9] used local block refinement technique in order to predict more precise result. Armaly et al [1] detailed experimental work with a step geometry having H/h = 1.94. Kim and Moin [10] computed using a second order space-time technique. The found a good agreement with experimental data for Re up to 500. The computed data started to deviate from the experiment from Re = 600. Durst et al [11] observed a second flow separation zone in two dimensional numerical simulation of a symmetric sudden expansion flow. Kaiktsis et al [12] observed that the unsteadiness was created by convective instability.

OBJECTIVE

The purpose of the study was to investigate the flow behavior through the backward facing step and to calculate flow separation and recirculation zone at different Re. In order to validate the result, the simulated data would have to compare with the experimental data [1].

NOMENCLATURE

u = Horizontal velocity component
v = Vertical velocity component
Re = Reynold's Number
x_r = First reattachment length
x_4 = first Separation length at the top face
x_5 = first reattachment length at the top face
h = step height
μ = Dynamic viscosity
ρ = Density

THEORY

The simulation is done by solving 2D Navier-Stoke equation for constant density flow. The continuity and momentum equation for a steady state 2D constant density flow are given as follows-

Continuity Equation,

$$\frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) = 0$$

For Constant density,

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

Momentum Equation,

X-Momentum:

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \vartheta\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right]$$

Y-Momentum:

$$u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \vartheta\left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right]$$

Here u, v are the horizontal and vertical velocity component of the flow. The boundary conditions are assumed as-

$$@\ wall: u = v = 0\ (no\ slip)$$

$$@\ inlet: u = Based\ on\ Re,\ \ v = 0$$

$$@\ outlet: \frac{\partial u}{\partial x} = 0; \qquad \frac{\partial v}{\partial y} = 0$$

NUMERICAL MODELING

Geometry

The geometry is developed based on the step size. An expansion ratio of 1:2 is assumed during the development of geometry. Other parameters are assumed as a function of h. Figure 1 shows the schematic of the geometry of a backward facing step. The step size is assumed a 5mm. To predict accurate flow behavior the downstream domain is assumed 20 times longer than the step.

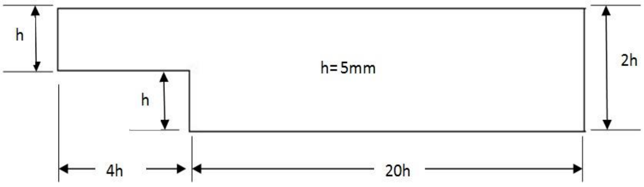


Figure 1. Geometry development

Mesh Generation

Around 0.18 million structured quadrilateral mesh is used within the entire domain. The mesh elements were clustered near the wall boundary (figure 2) in order to resolve the boundary layer. Maximum face size of the mesh is assumed as 0.025mm. The orthogonal quality of the mesh remains 0.98 and maximum aspect ratio is 1.43. A mesh independence test was done based on the flow separation length at Re = 100. Different mesh size is used to check mesh independence.

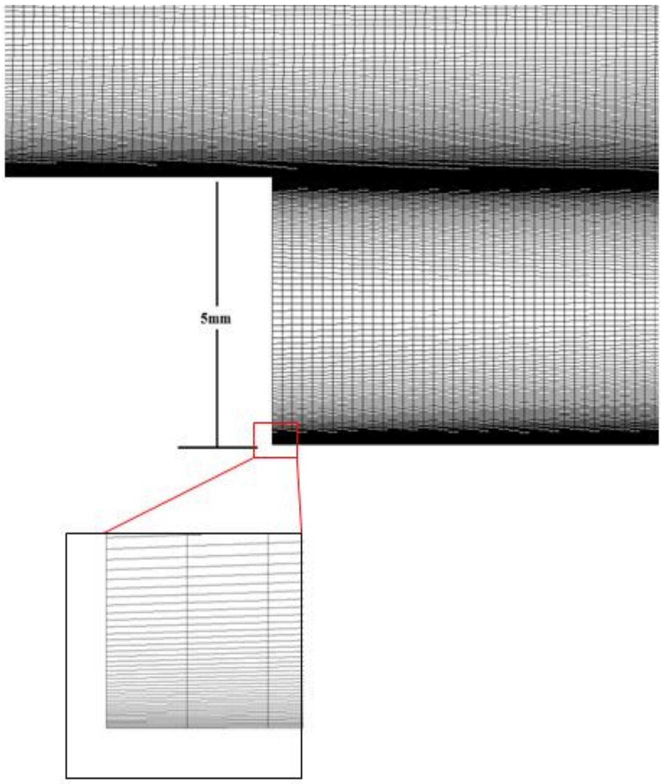


Figure 2. Mesh generation for the Model

Table 1 shows the mesh dependency on flow separation length. 5 different cases are checked for this keeping Re=100. From the table its clearly shown that the problem become mesh independent at mesh size of 400×450.

Table 1. Mesh dependency Test

Iteration	Mesh ($width \times Length$)	Xr/h
1000	50×100	4.8
1000	100× 200	4.5
1000	300× 400	3.9
1000	400× 450	3.8
1000	500× 500	3.8

Case setup and Solution

A 2D planner pressure based solver is used for the simulation. Water is considered as a working fluid. Inlet boundary condition is determined based on Re. For Re calculation the hydraulic diameter is assumed as 2h, where h is the step size.

$$Re = \frac{\rho v D_h}{\mu}$$

Standard least square discretization technique is used. A second order upwind solution method with SIMPLE solver is used. Re is set from 100 to 4000 as inlet condition. Pressure outlet is assumed constant. For convergence, the residue for continuity and velocity is set as 0.00001. Each solution is allowed to run 3000 iteration to met the minimum convergence criteria. To validate the CFD model, flow separation length (Xr/h) at different Re is compared with experimental data. Figure 3 shows the comparison of CFD results which have a good agreement with experimental data.

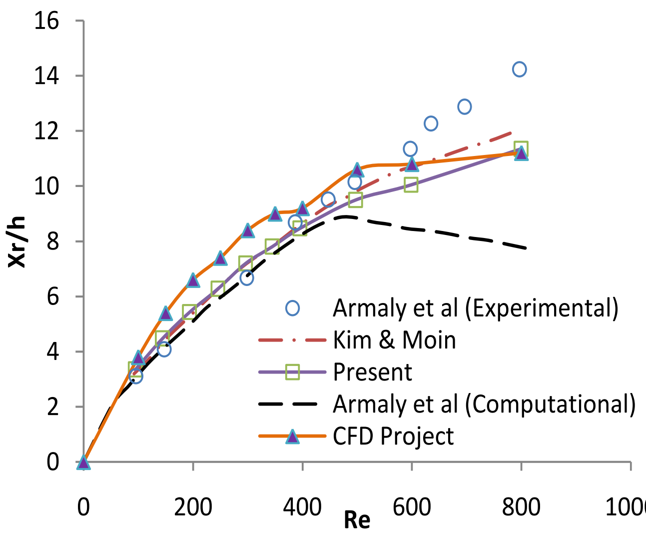
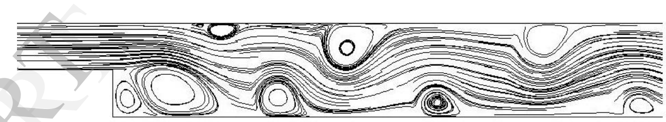


Figure 3. CFD result validation with experimental data

Results and Discussion

Horizontal and vertical velocity have been calculated for different Re ranging from 100 to 4000 in order to observe the flow behavior in Laminar , transition and turbulent regimes. Initially interval of Re is 50 between 100 to 400 after that interval is considered as 100 even more. For laminar flow at low Re (100-300) there is a single flow separation at the step and the length of the separation increases as Re increases. Figure 4(a) shows the flow separation stream lines at Re = 100. As Re increases separation occurs at different places. In this case second separation occurs at Re = 350. The separation length varies with Re and flow becomes more unstable and separated as Re increases from transition to turbulent. Figure 4(b) shows the flow separation at Re = 4000 with is considered as turbulent flow. Variation of the separation at different Re are shown in the APPENDIX.

(a)



(b)

Figure 4. Flow separation at different Re.
(a) Re = 100, (b) Re = 4000

Separation length at different position are compared with experimental data [1]. Figure 5 demonstrate the schematic of different separation length. Figure 6 shows the comparison for the X_1. It is observed that the result have good agreement with the experimental data at low Re (100-600). But as Re increases the deviation with experimental data also increases. Although they flows the same pattern. Identical result has been observed for X4 (figure 7) and X5 (figure 8).

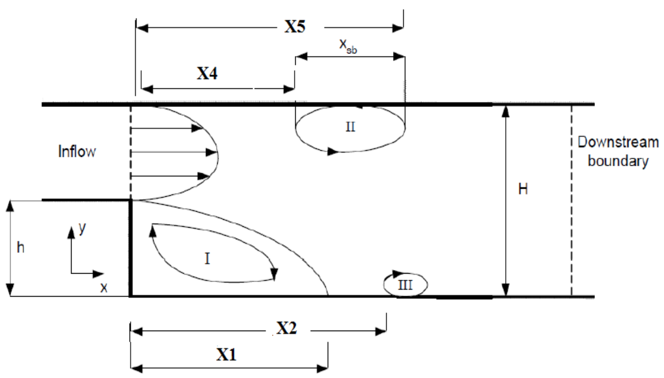


Figure 5. Schematic of different separation zone

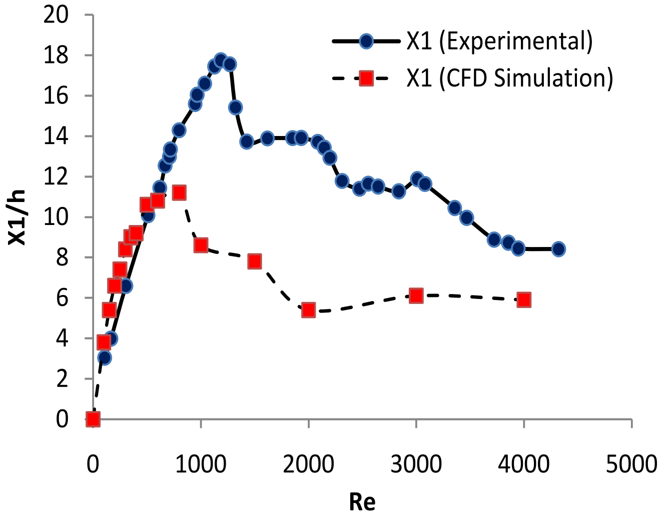


Figure 6. Comparison of X_1/h with experiment

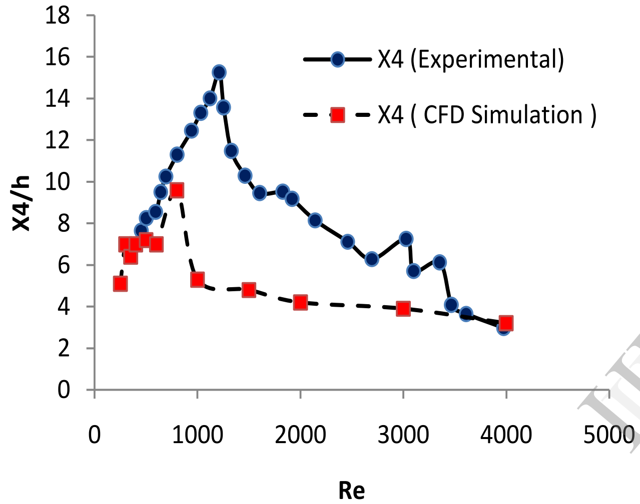


Figure 7. Comparison of X_4/h with experiment

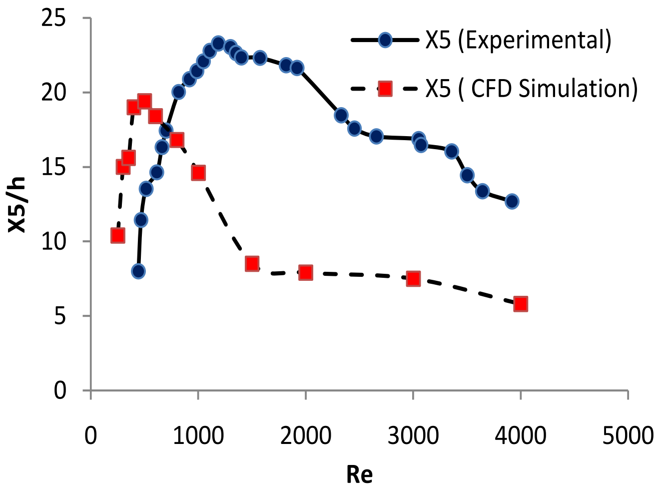


Figure 8. Comparison of X_5/h with experiment

Table 2 also validate the CFD result as it compares the reattachment length at different Re which agree with the experimental data as well. On the other hand figure 9 shows the percentage error for different recirculation zone. Figure 10 shows the combined X1, X4, X5 with the experimental data. The solid lines in figure 10 indicate the experimental data while the dashed lines are indicating the CFD result.

Table 2: Comparison of Reattachment length at different Re

Re	X_1 / h (Exp)	X_1 / h (CFD)	X_{sb} / h (Exp)	X_{sb} / h (CFD)
100	3.261	3.8		
150	4.379	5.4		
200	5.376	6.6		
250	6.230	7.4		
300	7.135	8.3		
350	7.791	9		5.4
400	8.448	9.2		7.6
500	9.475	10.4	6.384	9.2
600	10.015	10.8	8.713	12.2
800	11.338	11.2	11.776	9.8
1000	13.179	8.6	13.546	6.6

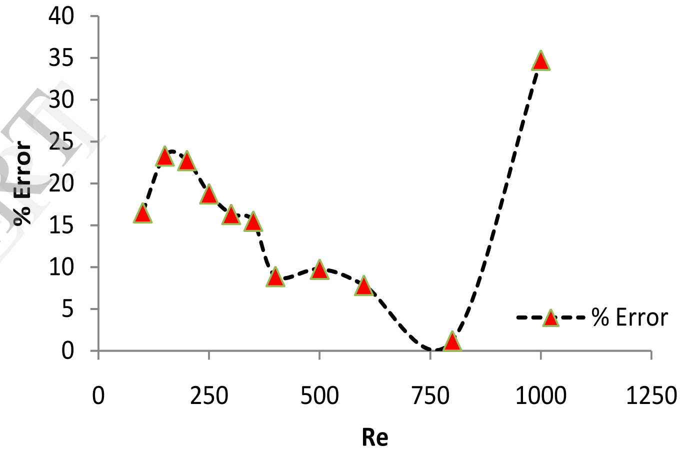


Figure 9. Percentage error for reattachment length

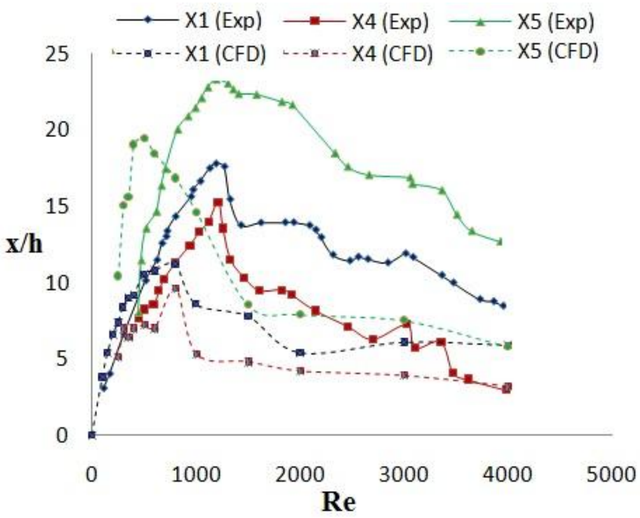


Figure 10. CFD data comparison with experiment

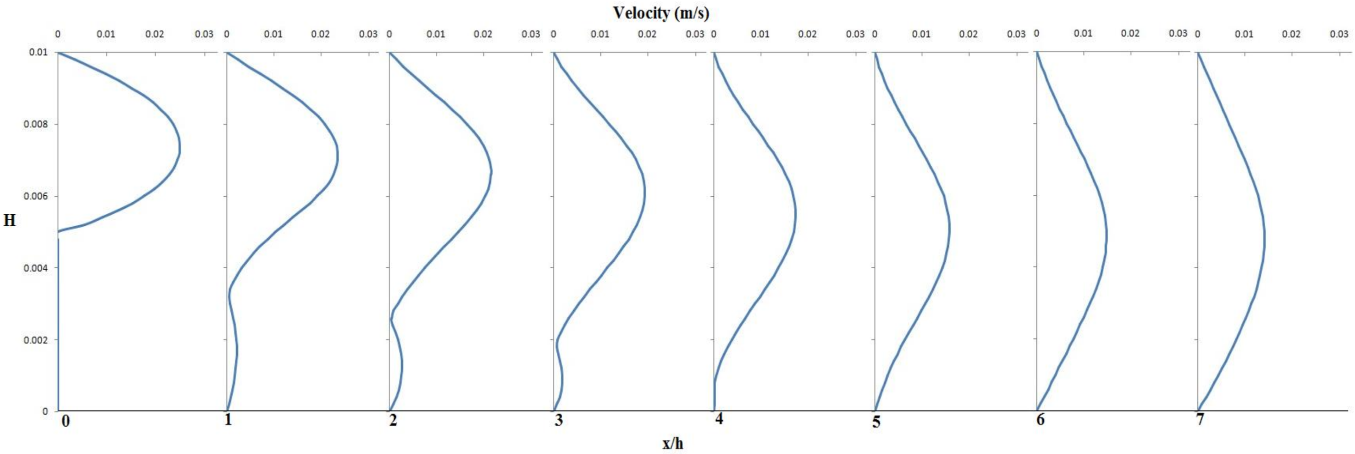
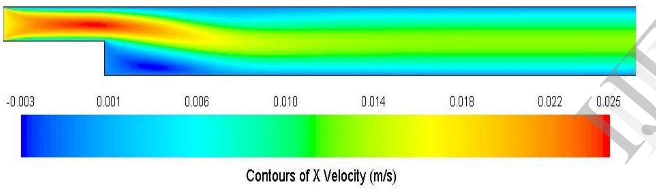
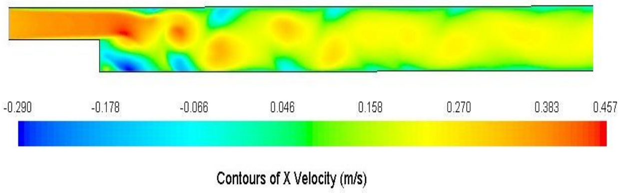


Figure 12. Horizontal velocity profile at different location (x/h) of the channel

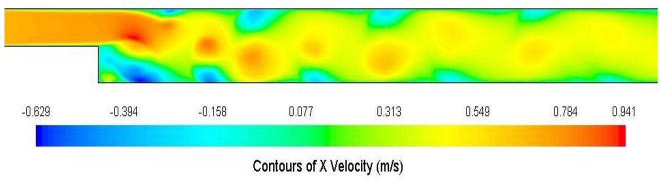
Horizontal and vertical velocity has also been calculated for different Re. Figure 12 shows the horizontal velocity profile at different location of the channel. It also shows the change of velocity due to flow separation and recirculation zone. Figure 13 and figure 14 show the velocity contour for Re = 100, Re = 2000 and Re = 4000 in order to demonstrate the laminar, transitional and turbulent flow for horizontal and vertical velocity component respectively. The figures also show the recirculation zones. Detail velocity contours are shown in the APPENDIX.



(a)

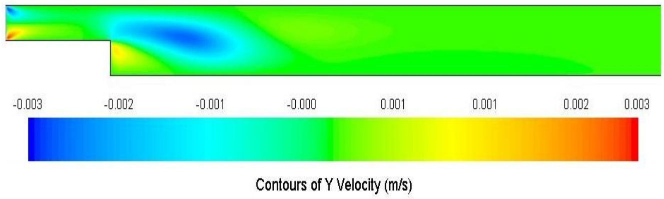


(b)

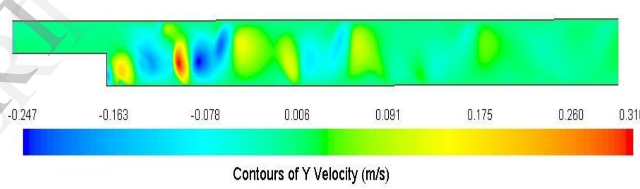


(c)

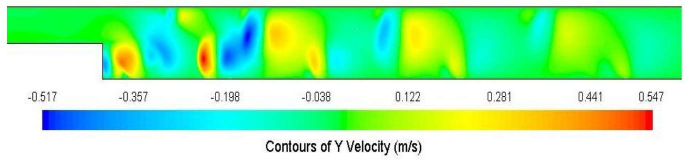
Figure 13. Horizontal velocity Contour
(a)Re=100, (b)Re=2000, (c)Re=4000



(a)



(b)



(c)

Figure 14. Vertical velocity component
(a) Re=100, (b) Re = 2000, (c) Re = 4000

Conclusion

CFD simulation has been done for a backward facing step for different Re in order to understand the flow behavior for laminar, transition and turbulent flow. To validate the result mesh independency test is done first. Then the separation length has been calculated and compared with the experimental data in order to verify the simulation result once again. Finally horizontal and vertical velocity contour and velocity distribution along the channel have been calculated and presented.

REFERENCE

[1] Armaly, B. F., Durst, F., Peireira, J. C. F., Scho¨nung, B., 1983, *Experimental and theoretical investigation of backward-facing step flow*, J. Fluid Mech.,**127**, pp. 473–496.

[2] Lee, Y. S. and Smith, L. C., 1986, *Analysis of power-law viscous materials using complex stream, potential and stress functions*, in Encyclopedia of Fluid Mechanics, vol. 1, Flow Phenomena and Measurement, ed. N. P. Cheremisinoff, pp. 1105–1154.

[3] Roache, P. J., 1972, *Computational Fluid Dynamics*, Hermosa, New Mexico, pp. 139–173.

[4] Taylor, T. D., and Ndefo, E., 1971, *Computation of viscous flow in a channel by the method of splitting*, Proc. of the Second Int. Conf. on Num. Methods in Fluid Dynamics, Lecture Notes in Physics, vol. **8**, pp. 356–364, Springer Verlag, New York.

[5] Durst, F., and Peireira, J. C. F., 1988, *Time-dependent laminar backwardfacing step flow in a two-dimensional duct*, ASME J. Fluids Eng., **110**, pp. 289–296.

[6] Alleborn, N., Nandakumar, K., Raszillier, H., and Durst, F., 1997, *Further contributions on the two-dimensional flow in a sudden expansion*, J. Fluid Mech., **330**, pp. 169–188.

[7] Brandt, A., Dendy, J. E., and Ruppel, H., 1980, *The multigrid method for semi-implicit hydrodynamic codes*, J. Comput. Phys., **34**, pp. 348–370.

[8] Hackbusch, W., 1985, *Multigrid Methods for Applications*, Springer, Berlin.

[9] Lange, C. F., Scha¨fer, M., and Durst, F., 2002, *Local block refinement with a multigrid flow solver*, Int. J. Numer. Methods Fluids **38**, pp. 21–41.

[10] Kim, J., and Moin, P., 1985, *Application of a fractional-step method to incompressible Navier-Stokes equations*, J. Comput. Phys., **59**, pp. 308–323.

[11] Durst, F., and Peireira, J. C. F., and Tropea, C., 1993, *The plane symmetric sudden-expansion flow at low Reynolds numbers*, J. Fluid Mech., **248**, pp. 567–581.

[12] Kaiktsis, L., Karniadakis, G. E., and Orszag, S. A., 1996, *Unsteadiness and convective instabilities in a two-dimensional flow over a backward-facing step*, J. Fluid Mech., **321**, pp. 157–187.

APPENDIX

Path line

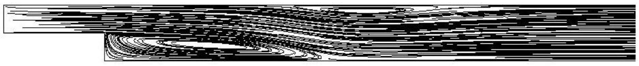
Re = 100

Re = 150

Re = 200

Re = 250

Re = 300

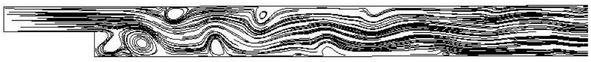


Re = 350

Re = 500

Re = 600

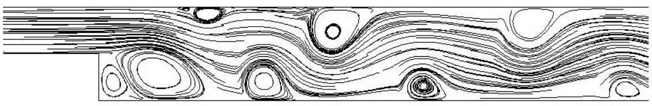
Re = 800



Re = 1500

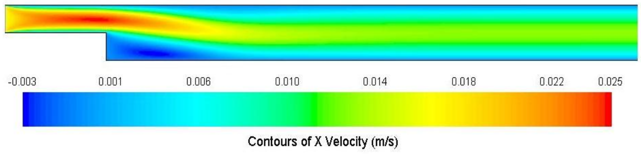
Re = 2000

Re = 3000

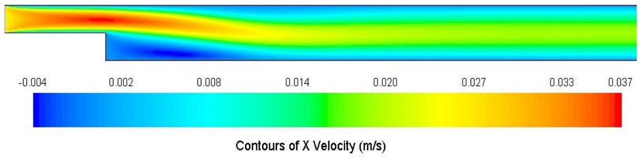


Re = 4000

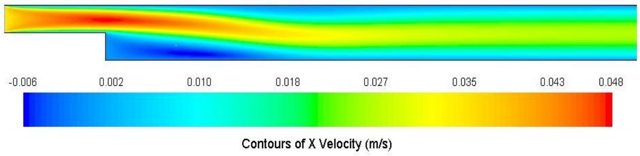
X-Velocity



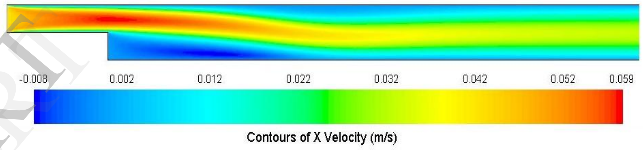
Re = 100



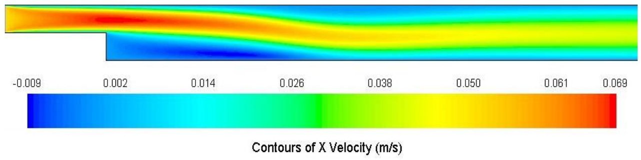
Re = 150



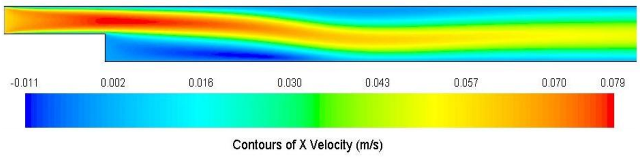
Re = 200



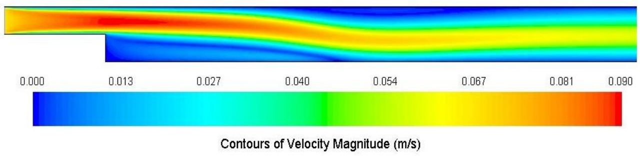
Re = 250



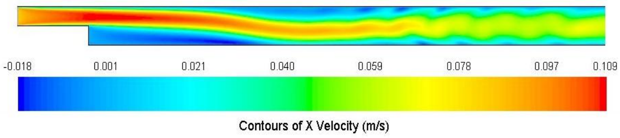
Re = 300



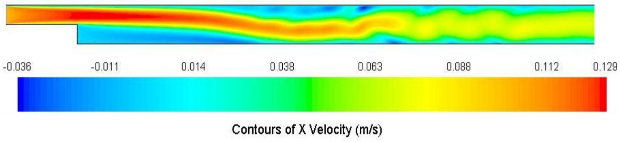
Re = 350



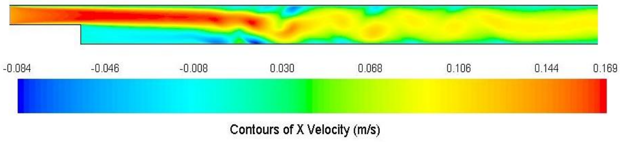
Re = 400



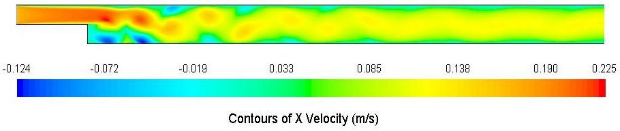
Re = 500



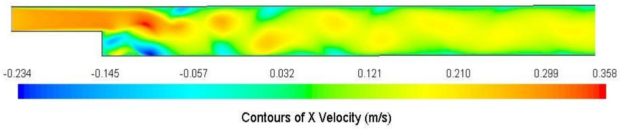
Re = 600



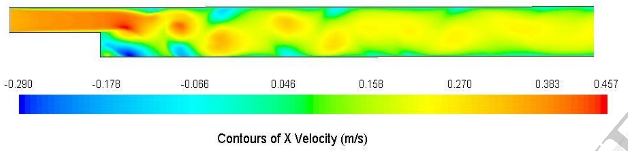
Re = 800



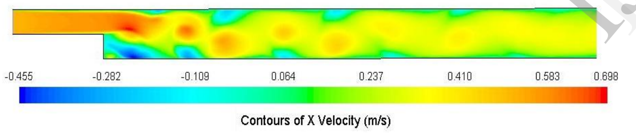
Re = 1000



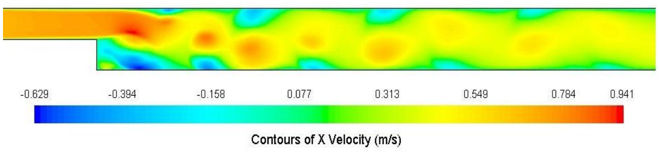
Re = 1500



Re = 2000

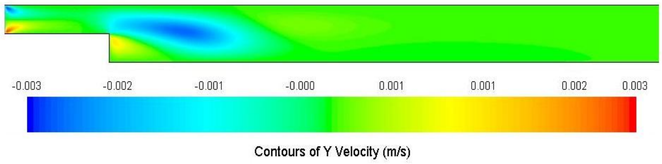


Re = 3000

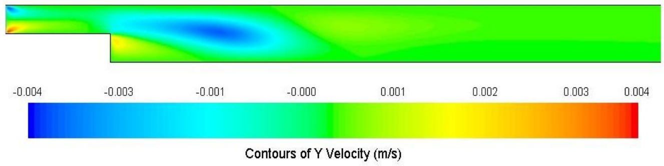


Re = 4000

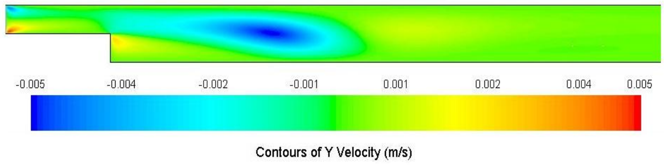
Y-Velocity



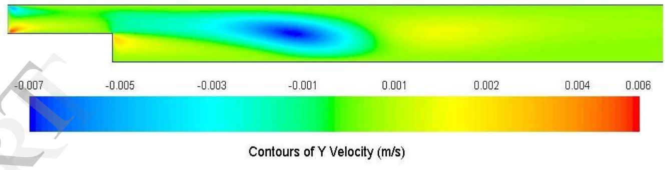
Re = 100



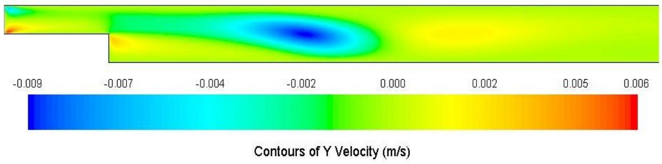
Re = 150



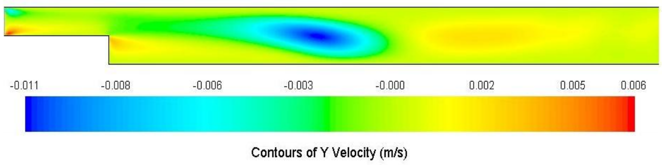
Re = 250



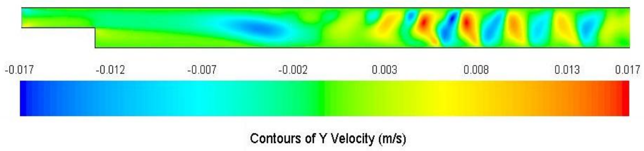
Re = 300



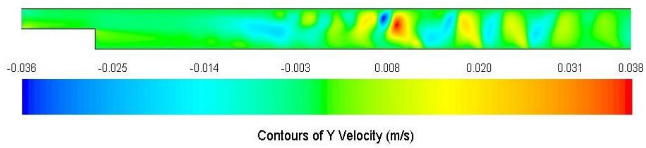
Re = 350



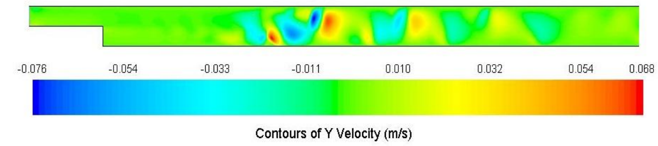
Re = 400



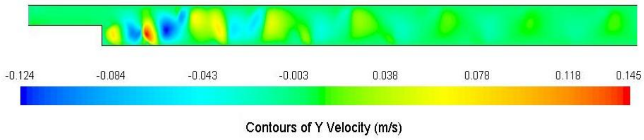
Re = 500



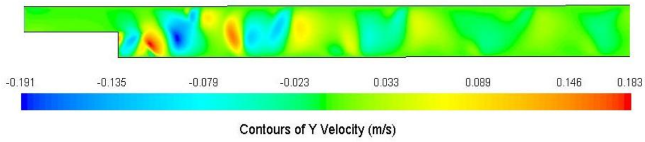
Re = 600



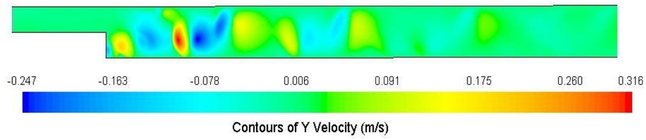
Re = 800



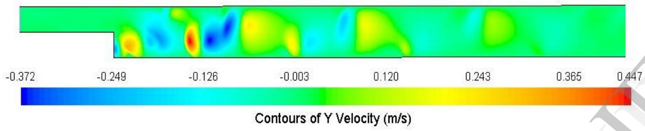
Re = 1000



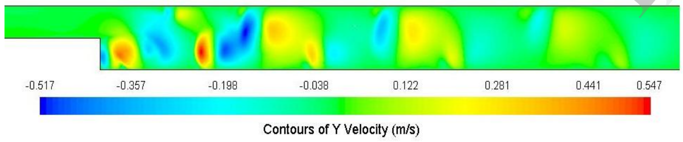
Re = 1500



Re = 2000

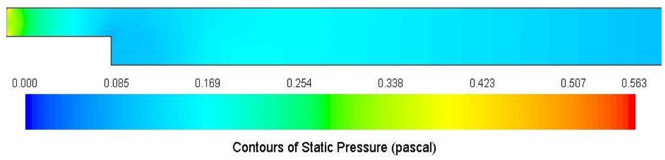


Re = 3000

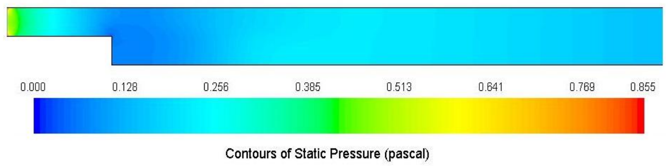


Re = 4000

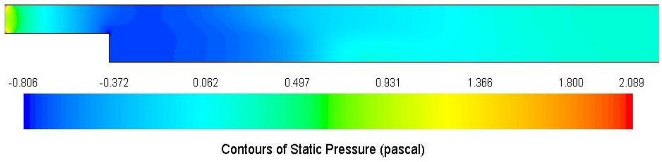
Pressure



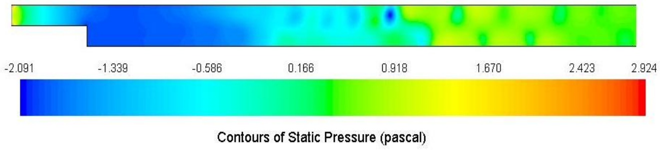
Re = 100



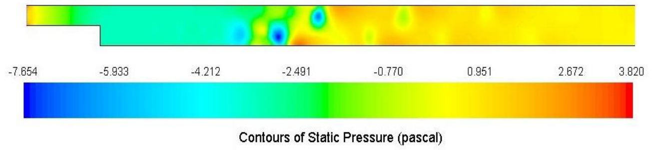
Re = 150



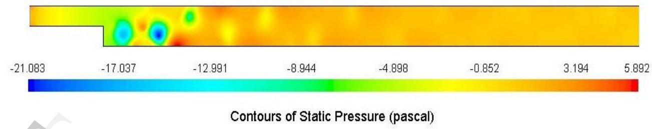
Re = 400



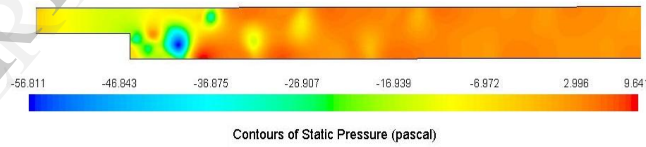
Re = 600



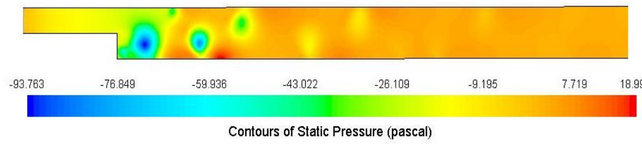
Re = 800



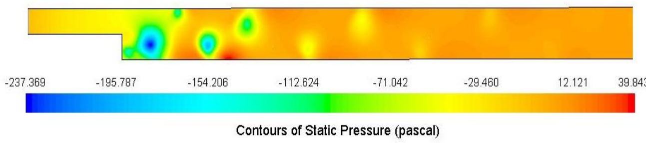
Re = 1000



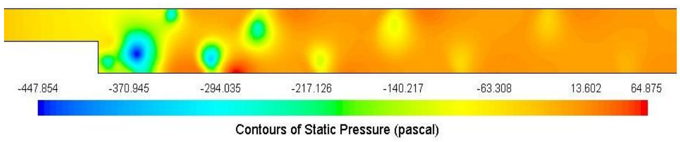
Re = 1500



Re = 2000



Re = 3000



Re = 4000